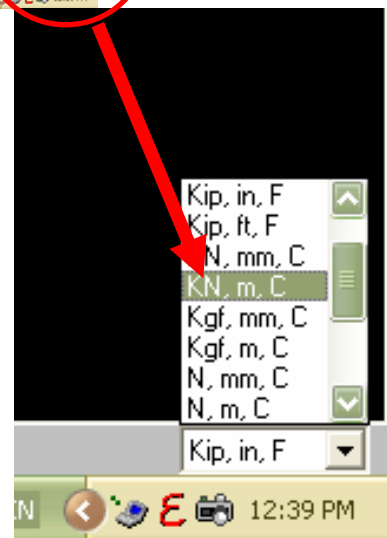
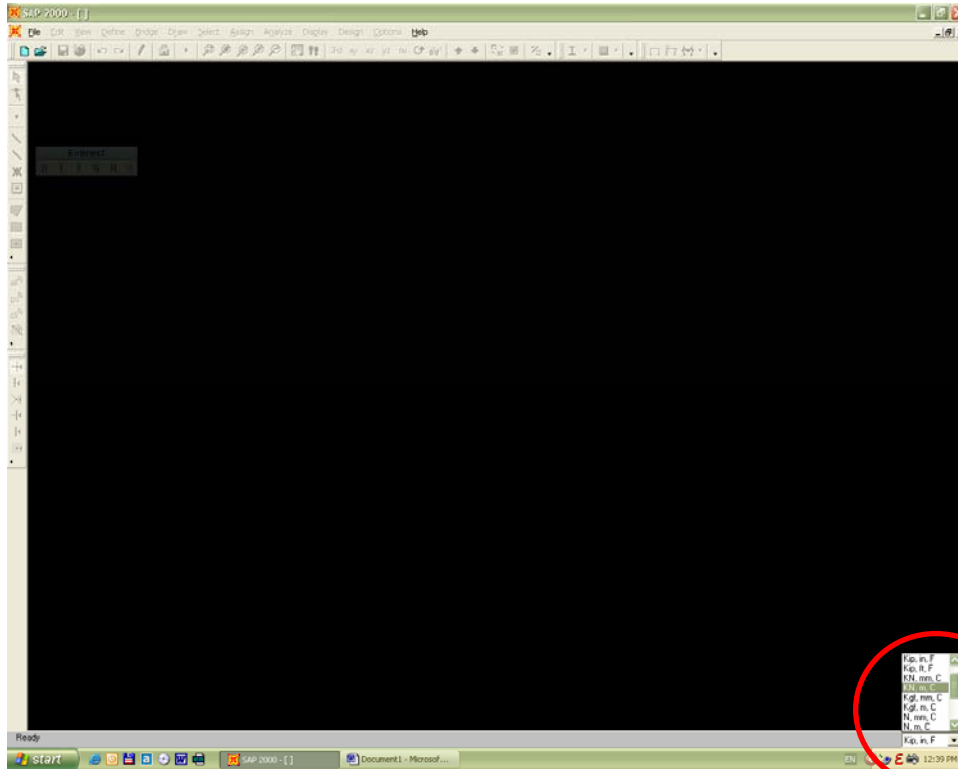


VISIT...

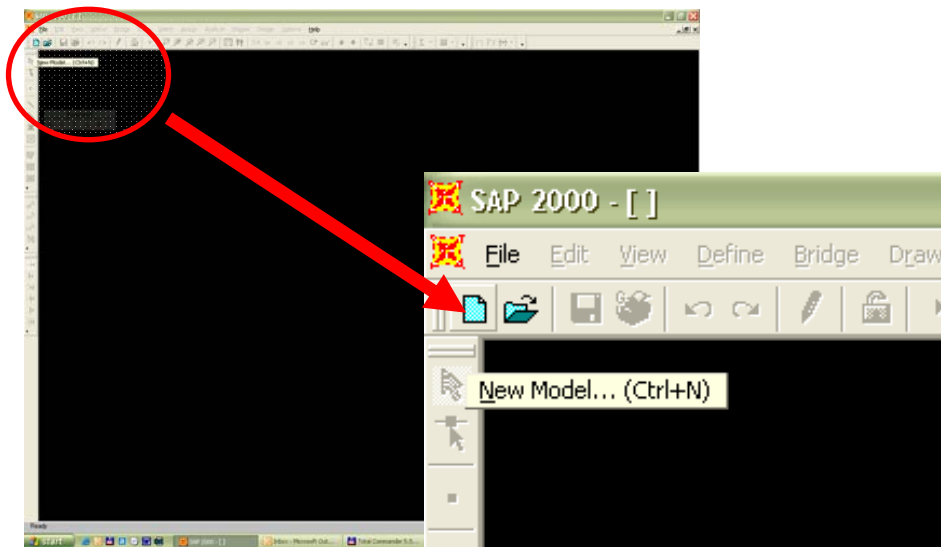
LANZAROTE
Caliente.COM

PROCEDURA DE INTRODUCERE A DATELOR ÎN SAP2000

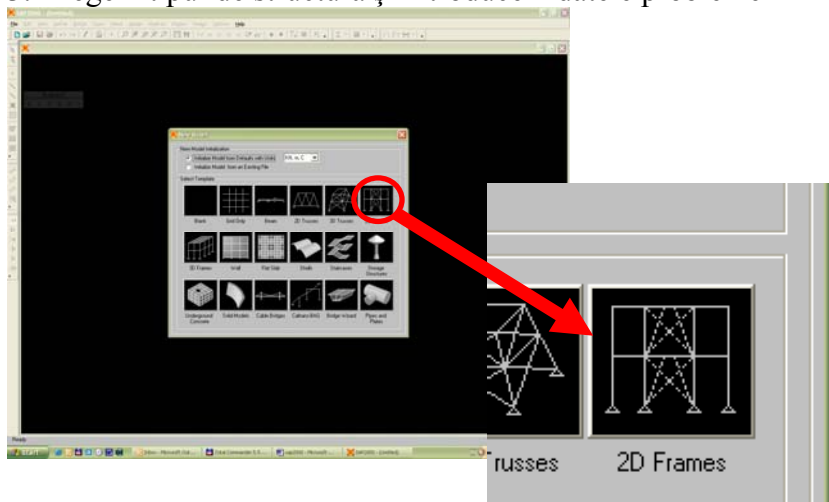
1. Setarea unităților de măsură din **Kip, in, F** în **KN, m, C**



2. Deschiderea unei planșe noi de lucru: **File → New Model**



3. Alegem tipul de structură și introducem datele problemei



2D Frames

2D Frame Type
Braced [Concentric]

Braced Frame [Concentric] Dimensions

Number of Stories	6	Story Height	4
Number of Bays	2	Bay Width	6

☐ Use Custom Grid Spacing and Locate Origin [Edit Grid...](#)

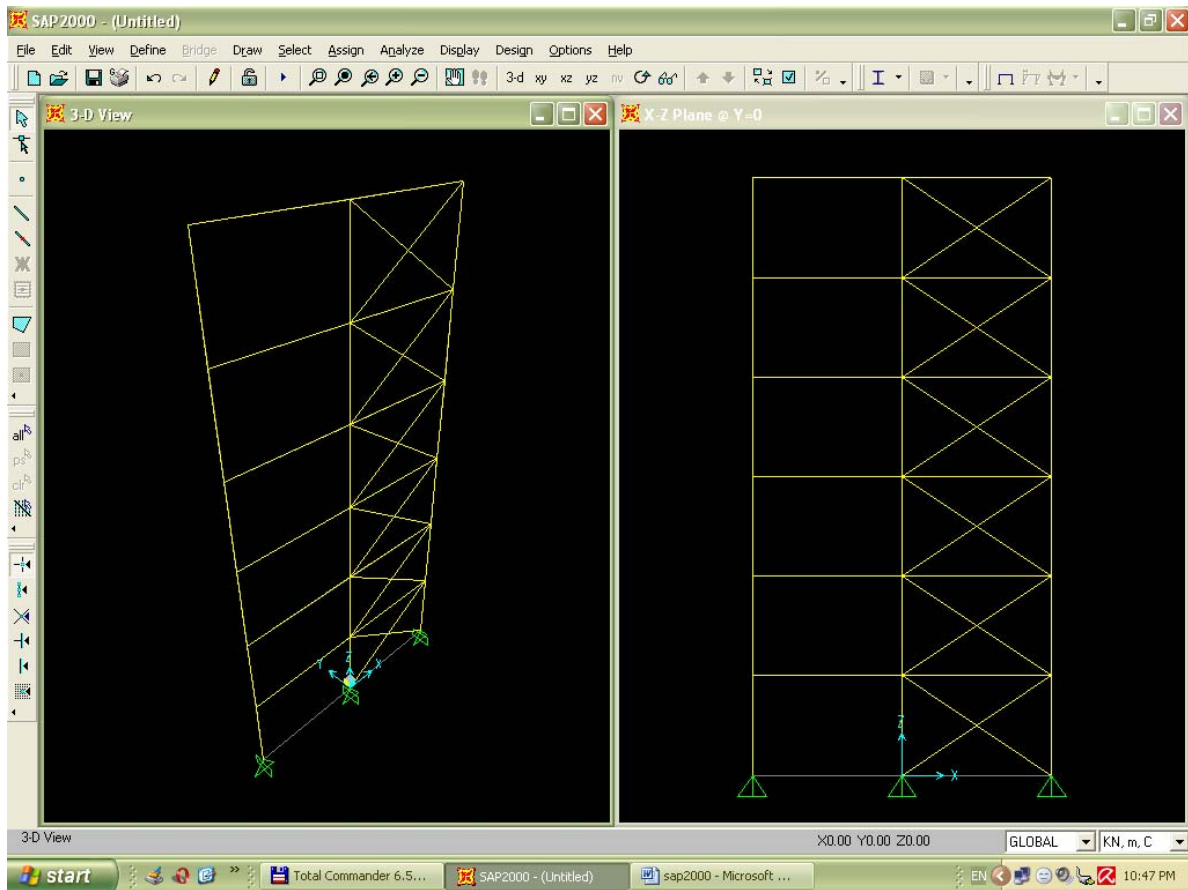
Section Properties

Beams	W18x35
Columns	W18x35
Braces	W18x35

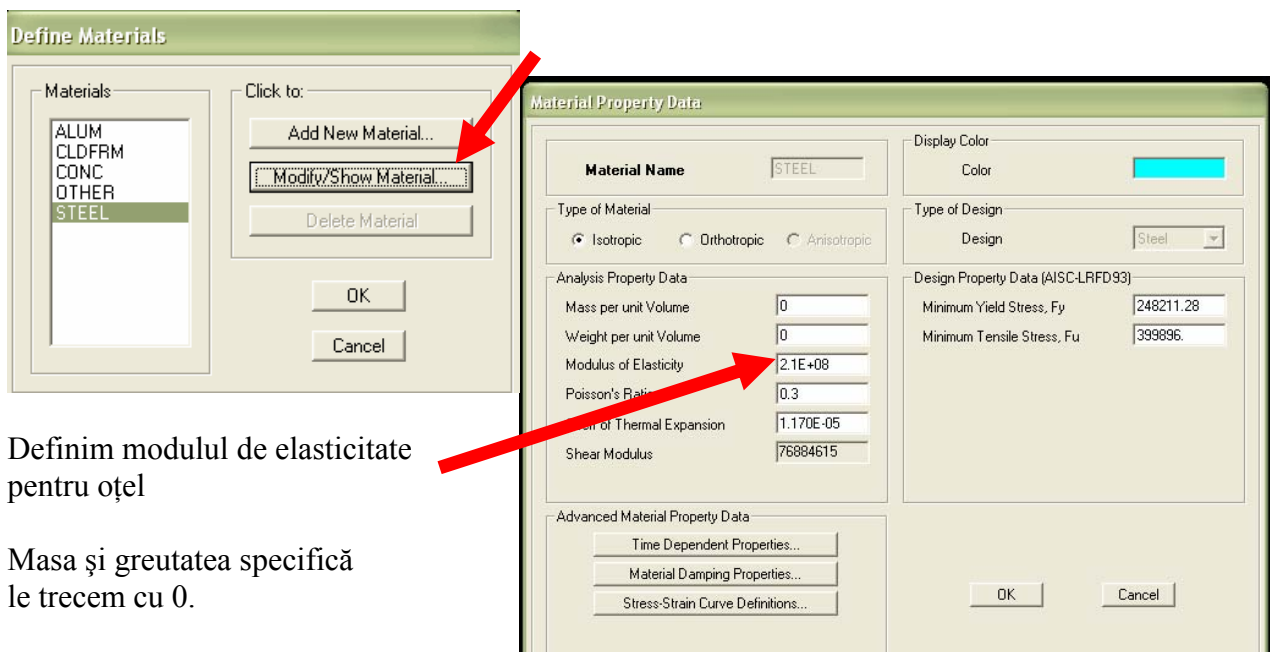
☒ Restraints

[OK](#) [Cancel](#)

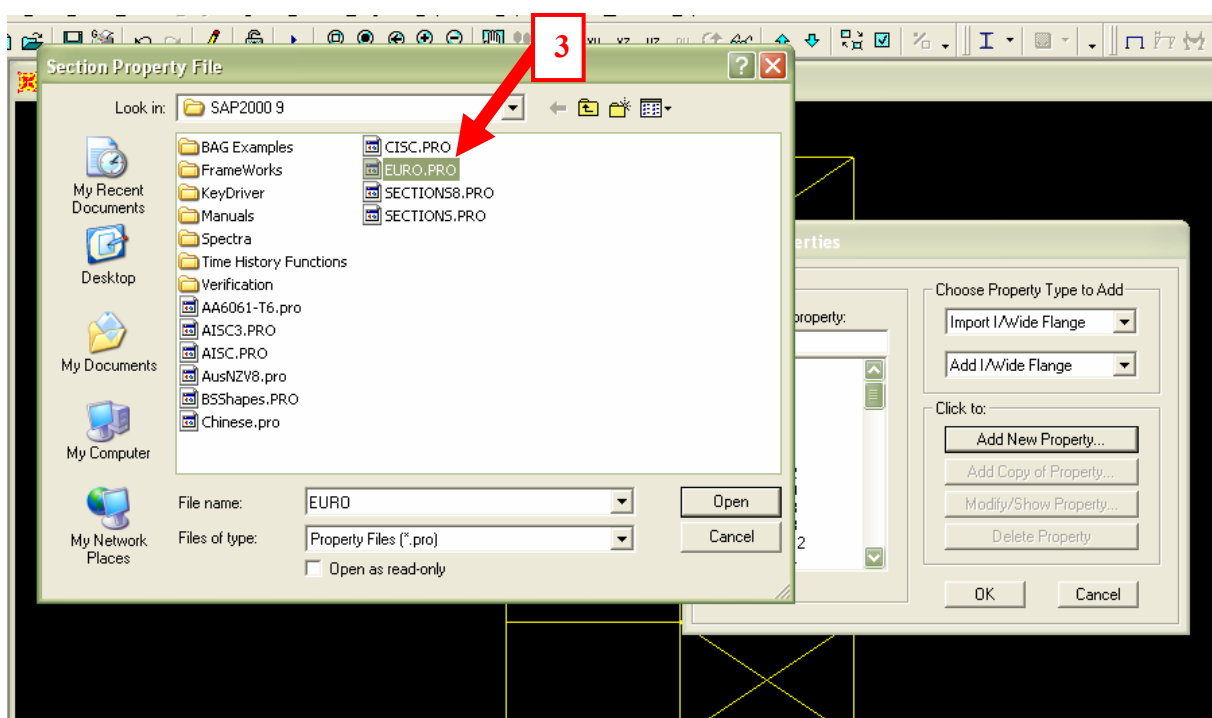
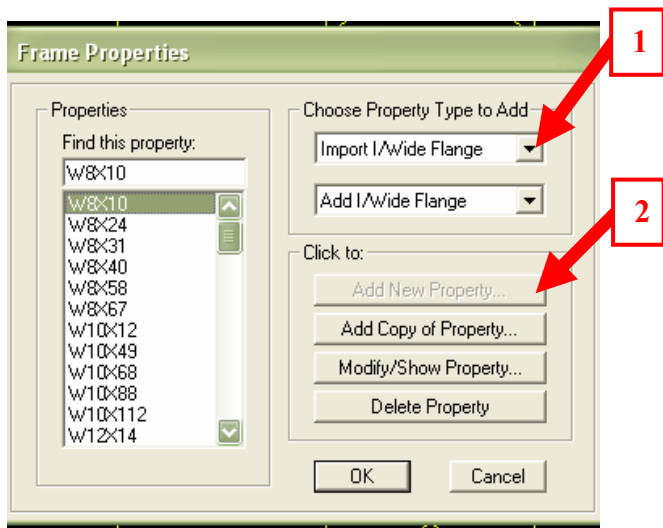
Rezultatul:



3. Definirea materialelor: **Define → Materials**

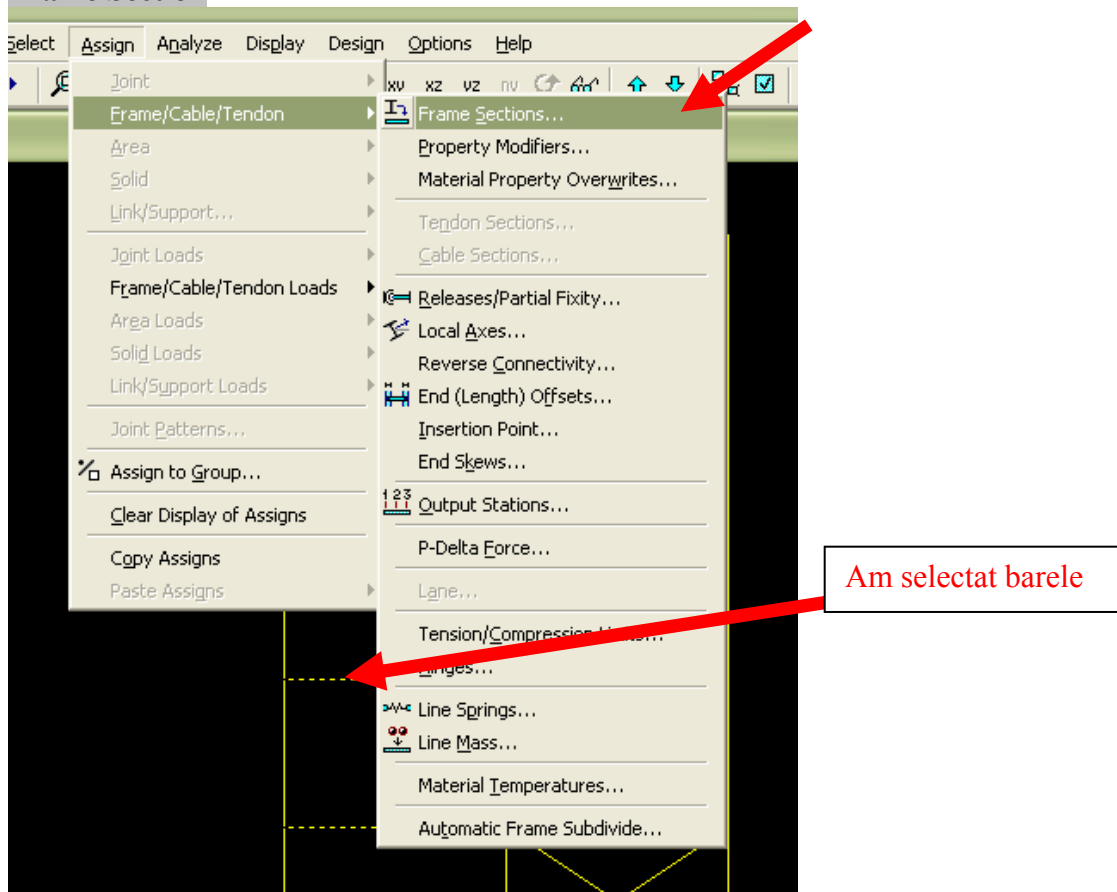


3. Definirea secțiunilor: **Define → Frame sections**

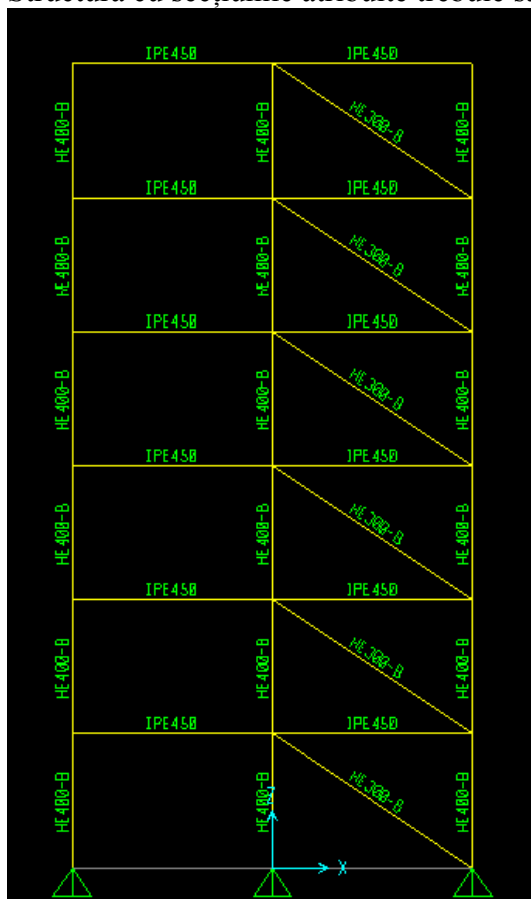


Alegem cu **CTRL** **HE 400B** pentru stâlpi, **IPE 450** pentru rigle și **HE 300B** pentru contravântuiri.

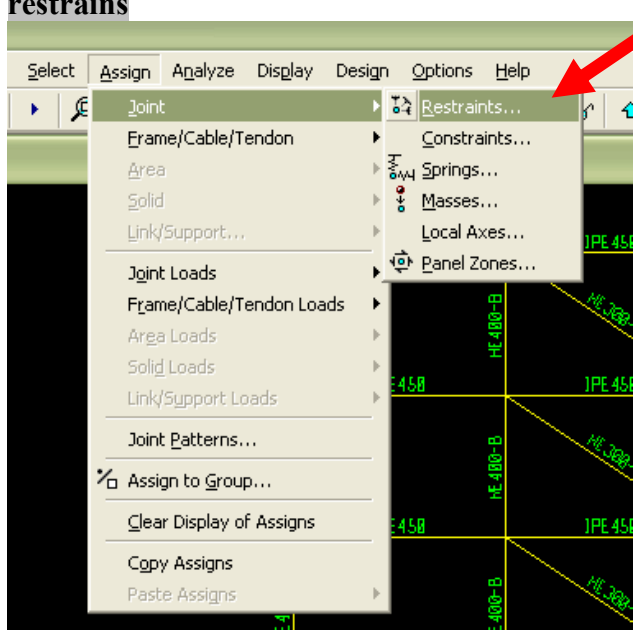
4. Atribuirea secțiunilor: selectăm barele și atribuim prin **Assign → Frame/Cable/Tendon→ Frame Section**



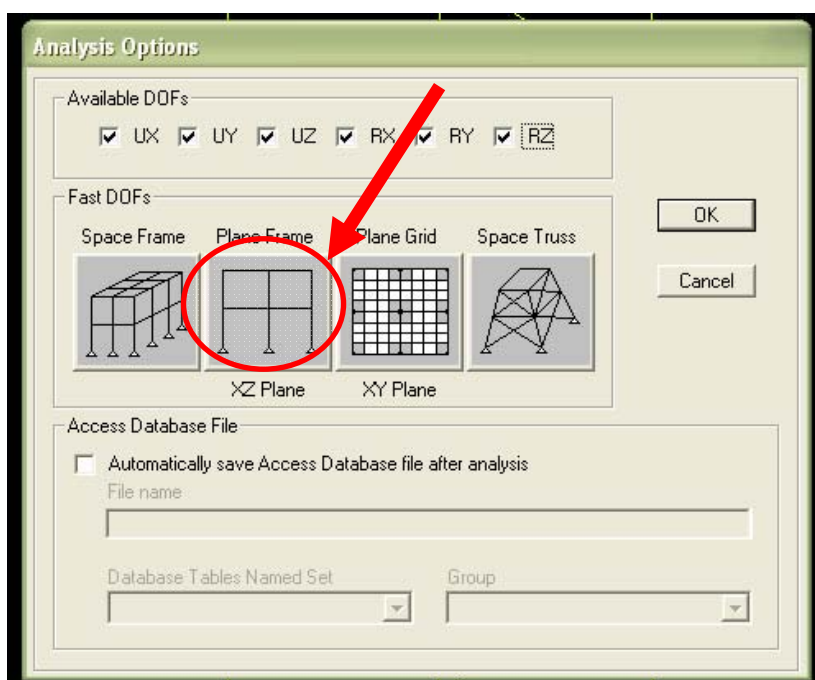
Structura cu secțiunile atribuite trebuie să arate în final așa:



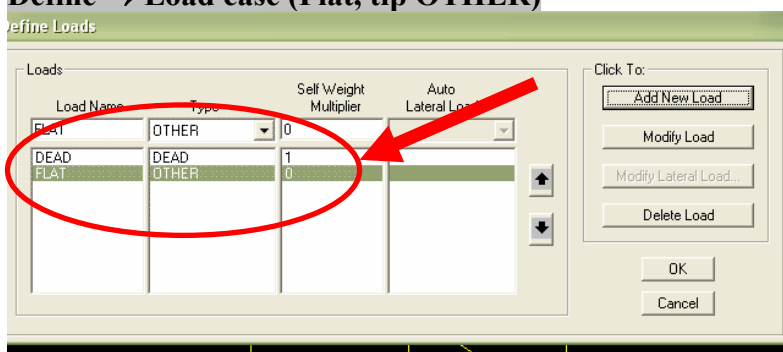
5. Atribuirea reazemelor: selectăm nodurile si atribuim reazeme blocate: **Assign → Joints→ restrains**



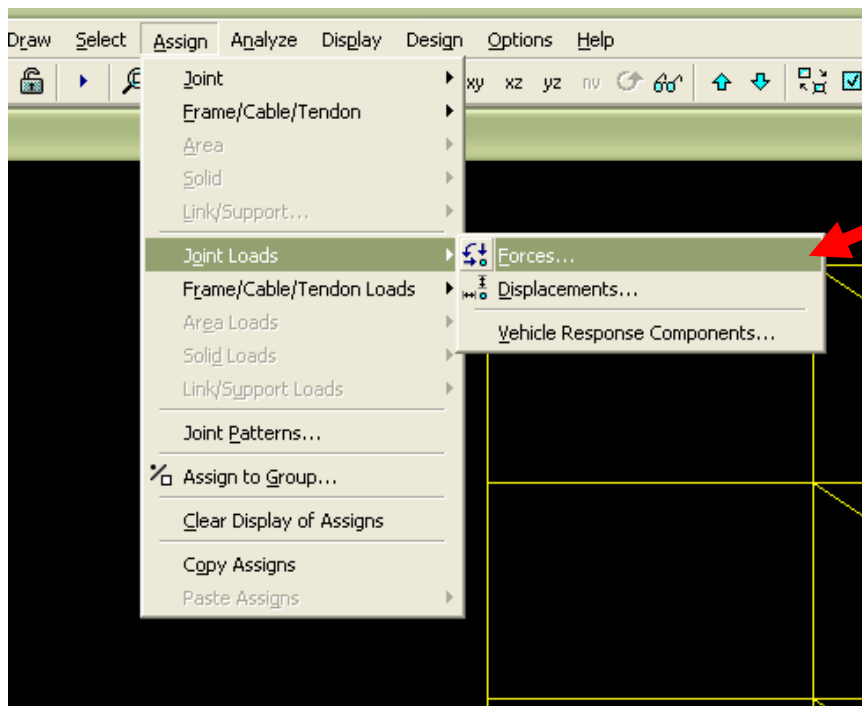
6. Definirea gradelor de libertate (structură plană):
Analyse → set analysis options→ plane frame (XZ)



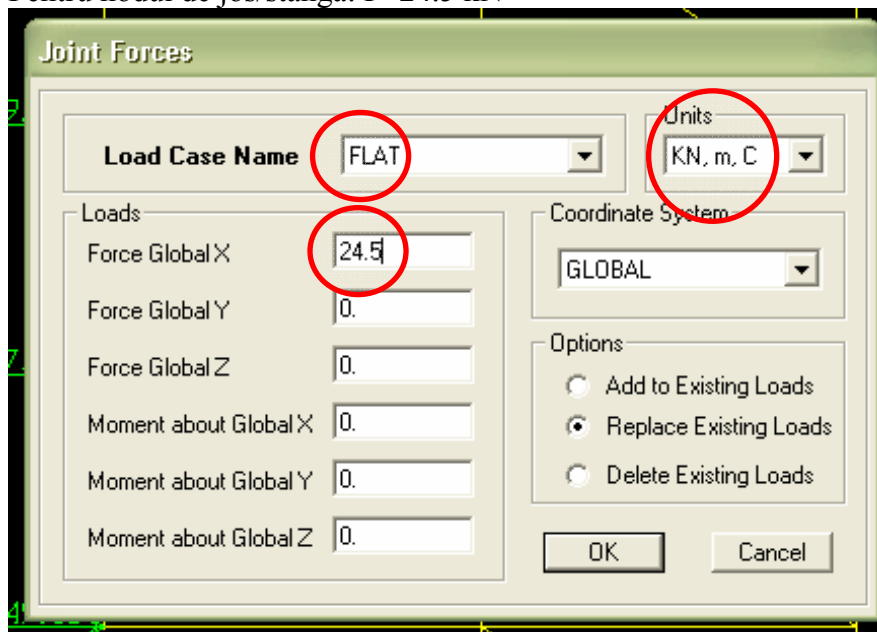
7. Definirea cazului de încărcare statică:
Define → Load case (Flat, tip OTHER)



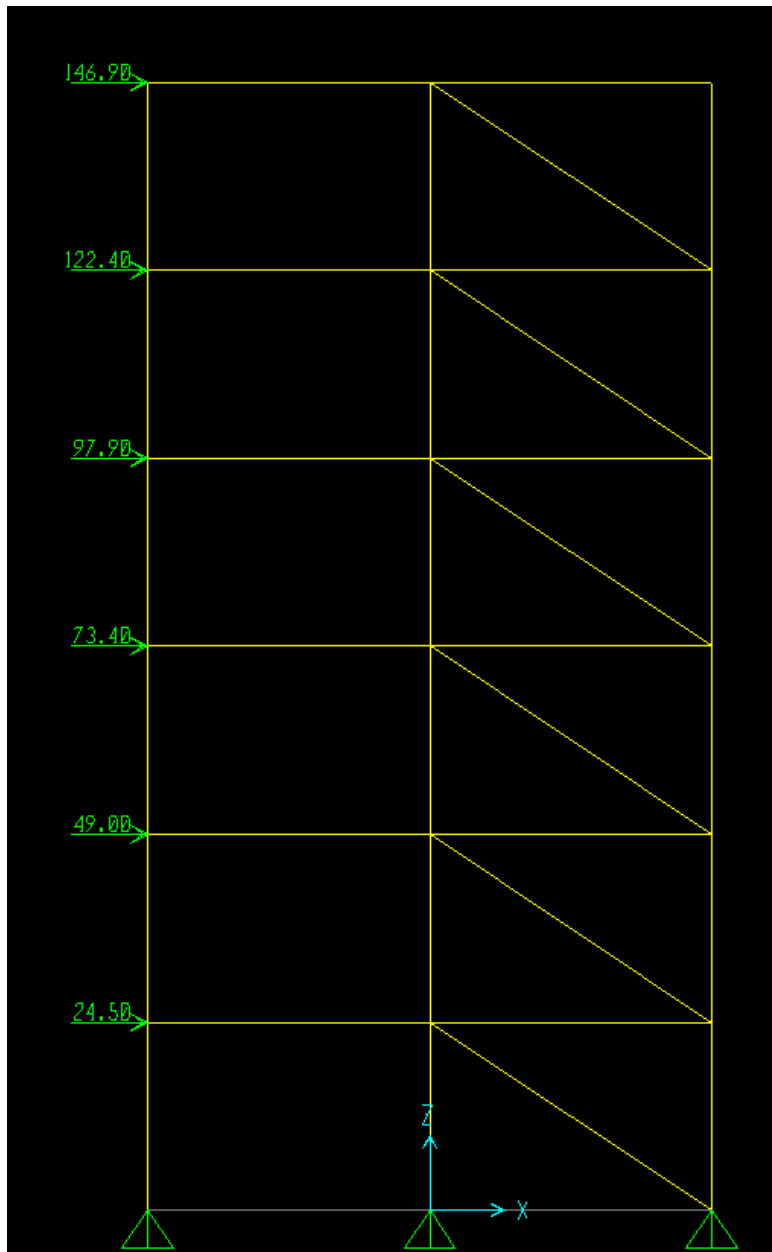
8. Definirea încărcări în noduri (forțe laterale): selectare nod
Assign → Joint Loads → Forces (în cazul de încărcare FLAT)



Pentru nodul de jos/stânga: $F=24.5$ kN

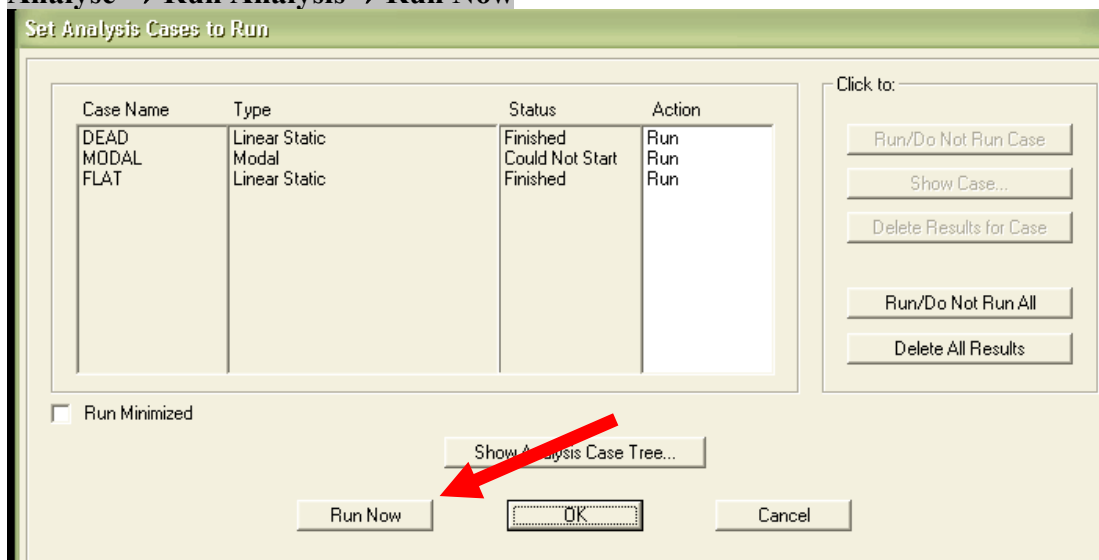


Structura cu forțele aplicate va fi:



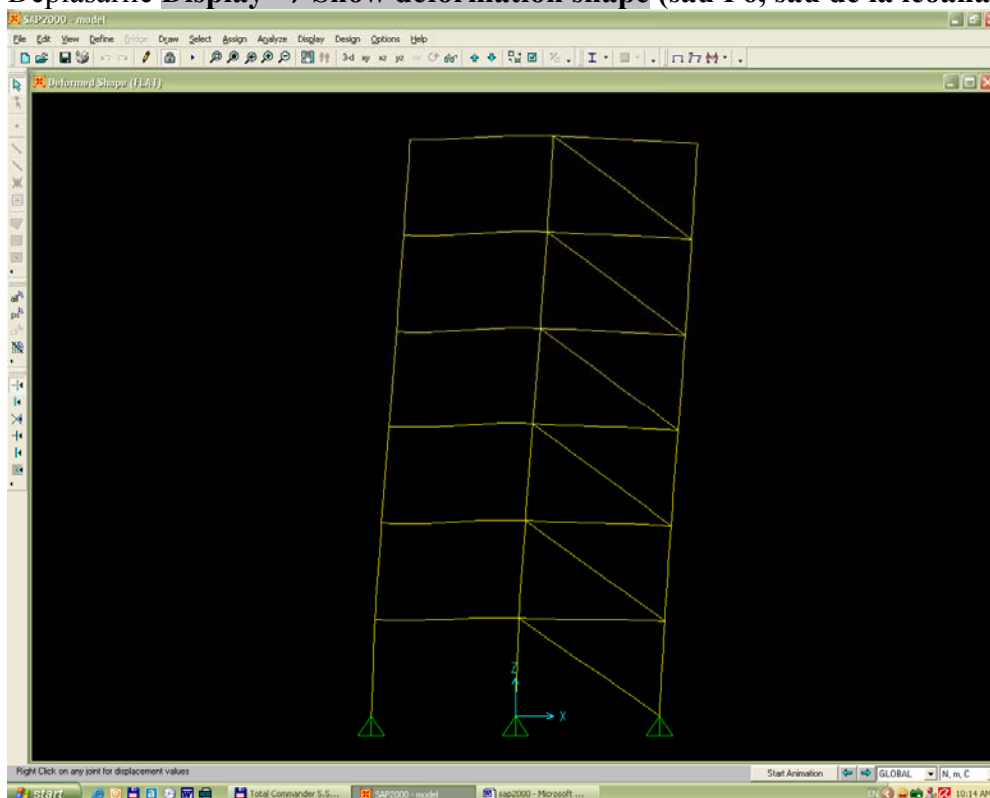
9. Efectuarea calculului:

Analyse → Run Analysis → Run Now



10. Vizualizarea rezultatelor:

Deplasările **Display → Show deformation shape** (sau F6, sau de la iconă )



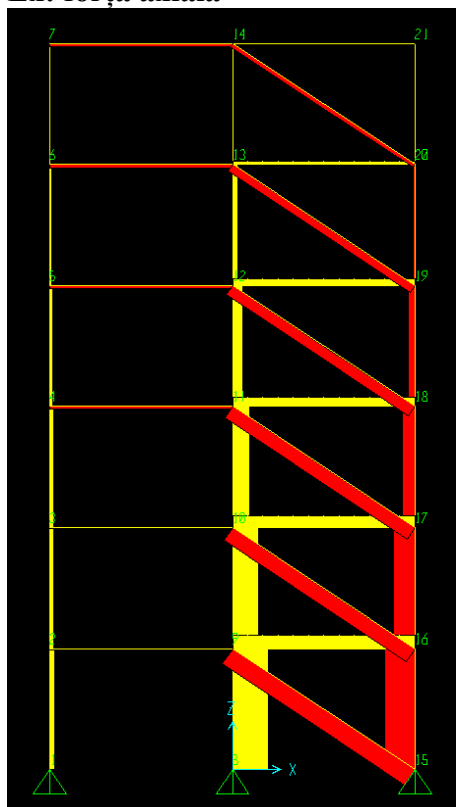
Eforturile: **Display → Show forces/stresses → Frames/Cables**

Pentru moment: **Moment 3-3**

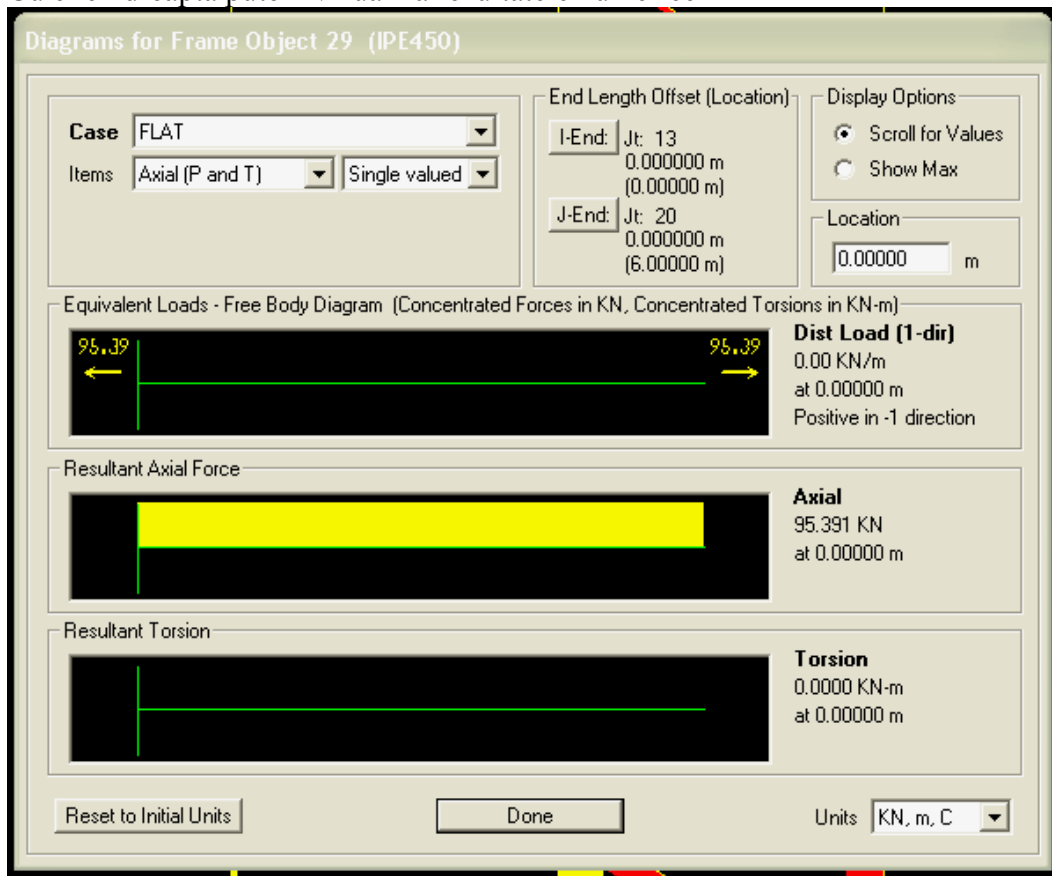
Pentru forța axială: **Axial force**

Pentru forța tăietoare: **Shear 2-2**

Ex: forța axială

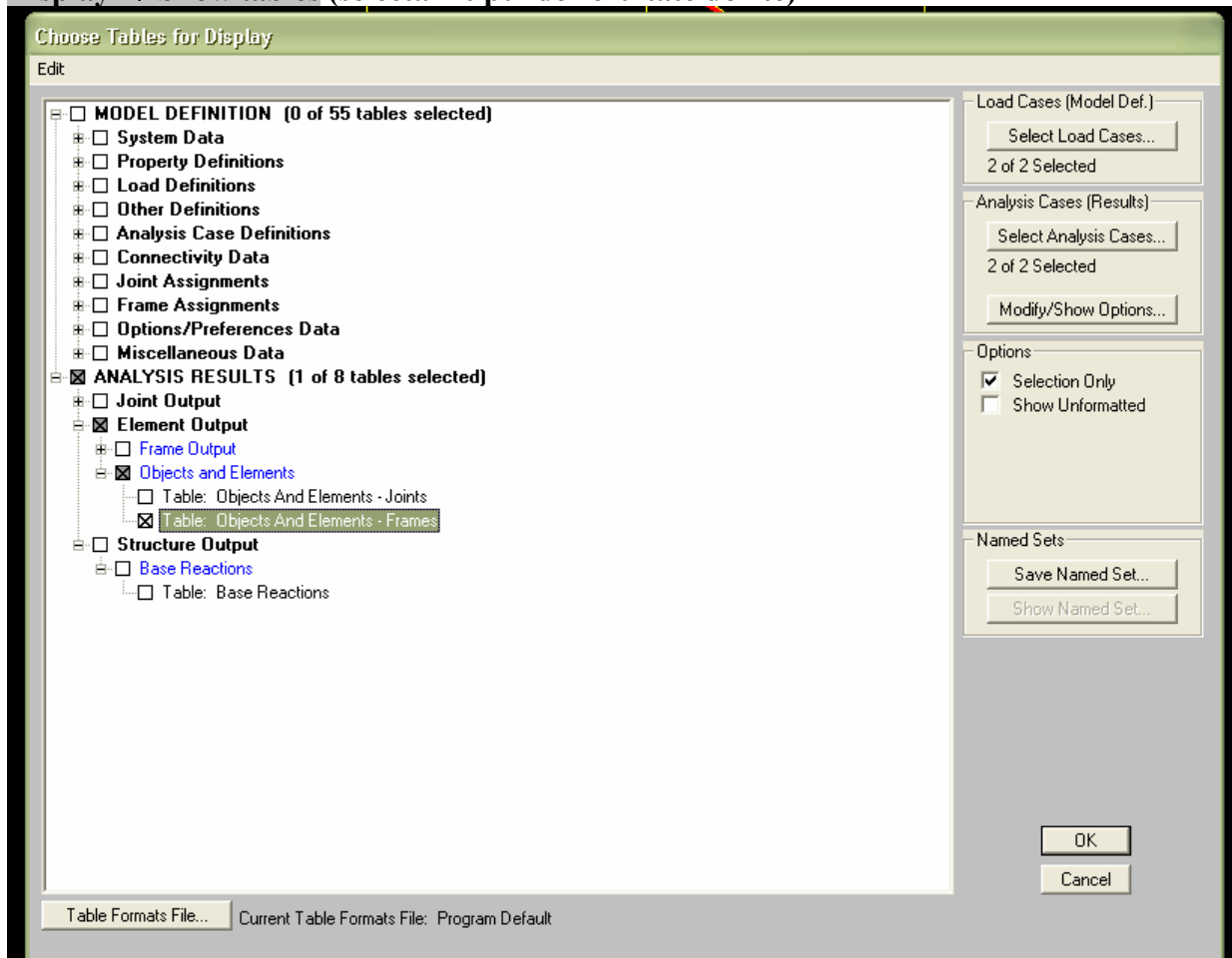


Cu click dreapta putem vizualiza rezultatele numerice



Vizualizarea rezultatelor, tabelar: selectăm elementele

Display → Show tables (selectăm tipul de rezultate dorite)



11. Procedura de introducere a datelor pentru analiză modală/spectrală:

ANALIZA MODALĂ

Întâi deblocăm modelul de la



- mase: selectăm nodurile **Assign → Joints → Masses**

- definire analiză modală

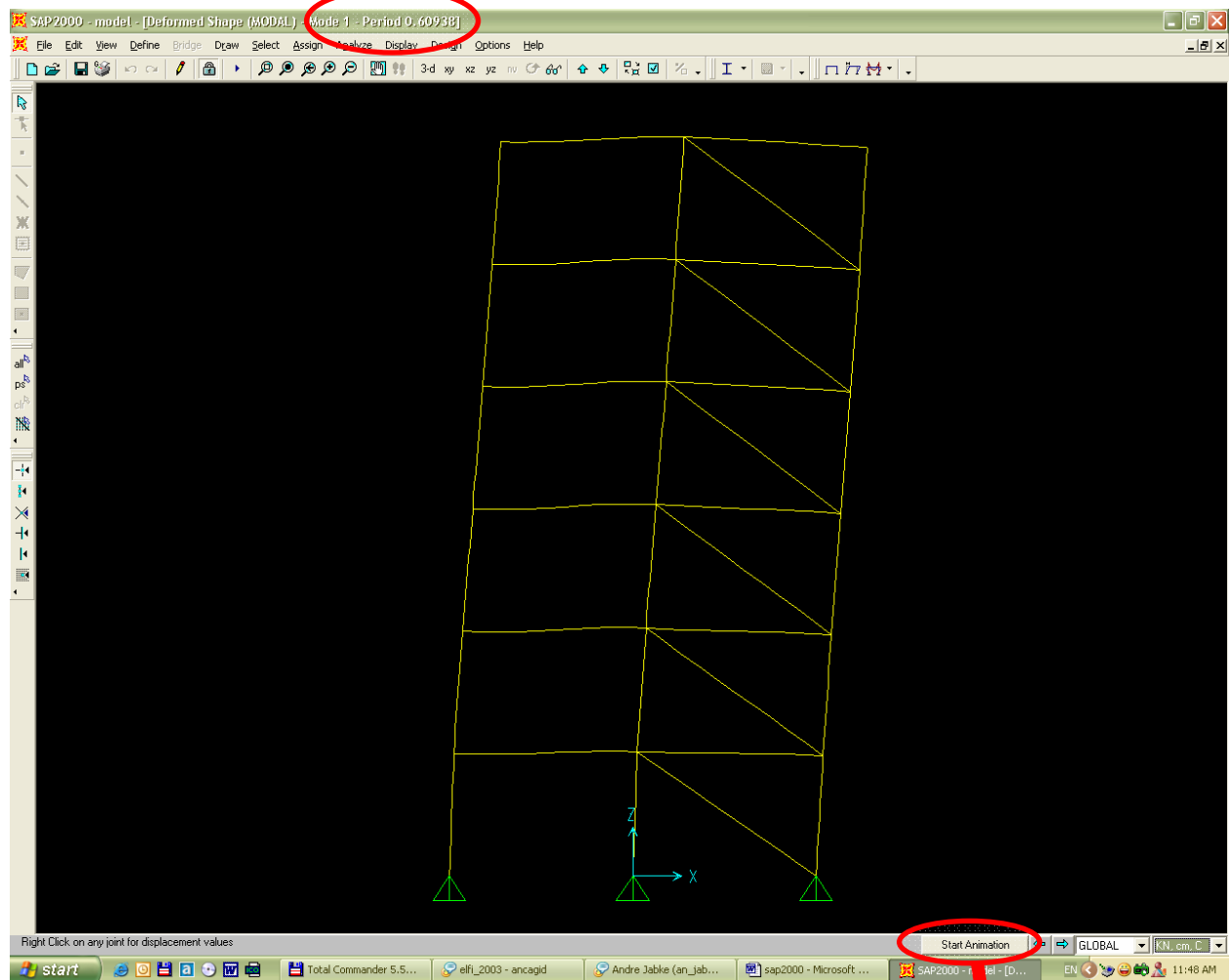
Define → Analysis Cases → Modal

Efectuare analiză: RUN 

Vizualizarea rezultatelor:

ODAL) - Mode 1 - Period 0.60938]

ex.: deplasări



modelul poate fi și animat



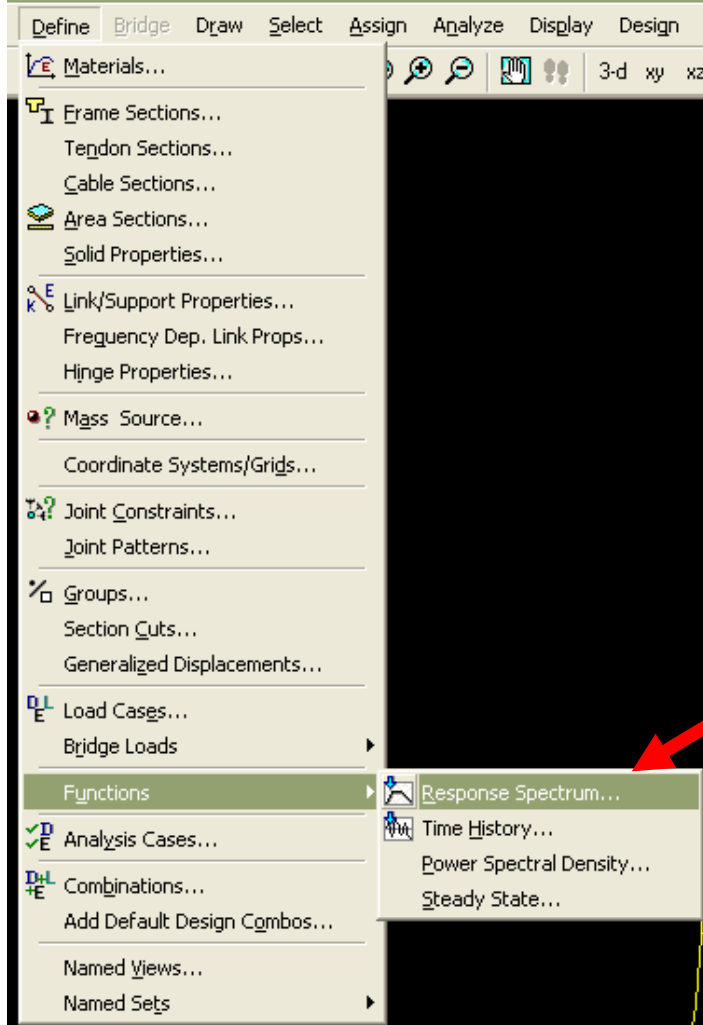
ANALIZA SPECTRALĂ

Întâi deblocăm modelul de la



Definirea spectrului

Define → Functions → Respons Spectrum (User Spectrum, Add new function)



Introducem datele dorite

Response Spectrum Function Definition

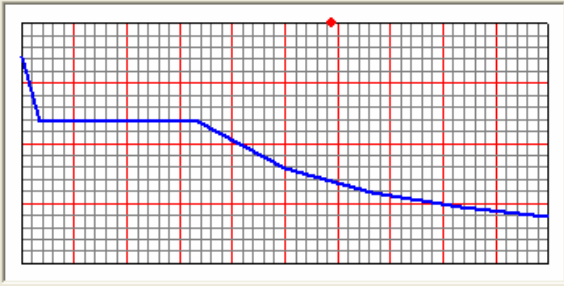
Function Name: UNIFRS

Define Function

Period	Acceleration
3	0.63
0.	2.75
0.1	1.89
1.	1.89
1.5	1.26
2.	0.945
2.5	0.756
3.	0.63

Add
Modify
Delete

Function Graph



Display Graph (0.5888, 1.)

OK Cancel

Definim calculul spectral

Define → Analysis Cases → Add new → Respons Spectrum

Analysis Case Data - Response Spectrum

Analysis Case Name: ploiesti Set Def Name

Analysis Case Type: Response Spectrum

Modal Combination: ☒ CQC ☐ SRSS ☐ ABS ☐ GMC ☐ 10 Pct ☐ Dbl Sum

GMC f1: GMC f2:

Directional Combination: ☒ SRSS ☐ ABS

ABS Scale Factor:

Modal Analysis Case: Use Modes from this Modal Analysis Case: MODAL

Loads Applied

Load Type	Load Name	Function	Scale Factor
Accel	U1	UNIFRS	1
Accel	U1	UNIFRS	1

Add
Modify
Delete

☐ Show Advanced Load Parameters

Other Parameters: Modal Damping: Constant at 0.05 Modify/Show...

OK Cancel

Efectuare analiză: RUN



Și vizualizăm după metodele descrise mai sus